

**CFD SIMULATION OF HEAT TRANSFER IN SHELL AND TUBE HEAT
EXCHANGER**

KHAIRUN HASMADI OTHMAN

**A thesis submitted in fulfillment for the award of the Degree of Bachelor in
Chemical Engineering (Gas Technology)**

**Faculty of Chemical and Natural Resources Engineering
Universiti Malaysia Pahang**

APRIL 2009

ABSTRACT

Computational Fluid Dynamic (CFD) is a useful tool in solving and analyzing problems that involve fluid flows, while shell and tube heat exchanger is the most common type of heat exchanger and widely use in oil refinery and other large chemical processes because it suite for high pressure application. The processes in solving the simulation consist of modeling and meshing the basic geometry of shell and tube heat exchanger using the CFD package Gambit 2.4. Then, the boundary condition will be set before been simulate in Fluent 6.2 based on the experimental parameters. Parameter that had been used was the same parameter of experimental at constant mass flow rate of cold water and varies with mass flow rate at 0.0151 kg/s, 0.0161 kg/s and 0.0168 kg/s of hot water. Thus, this paper presents the simulation of heat transfer in shell and tube heat exchanger model and validation to heat transfer in Shell and Tube Heat Exchanger Studies Unit (Model HE 667) that been used in UMP's chemical engineering laboratory. The CFD model is validated by comparison to the experimental results within 15% error.

ABSTRAK

Computational Fluid Dynamic (CFD) merupakan perisian yg sangat berguna dalam menyelesaikan dan menganalisa masalah yang melibatkan aliran bendalir, manakala 'shell and tube heat exchanger' merupakan jenis heat exchanger yang paling kerap digunakan di kilang pemprosesan minyak dan pemprosesan bahan kimia yang besar kerana sesuai digunakan untuk applikasi pada tekanan tinggi. Proses yang terlibat dalam menyelesaikan simulasi ini melibatkan 'modeling' dan 'meshing' geometri utama 'shell and tube heat exchanger' dengan menggunakan pakej CFD-Gambit 2.4. Kemudian, pembolehubah yang digunakan ditetapkan di dalam pakej Gambit sebelum diekspor ke pakej Fluent berdasarkan pembolehubah yang sama digunakan di dalam eksperimen. Pembolehubah yang digunakan ialah pembolehubah air sejuk yang malar dan air sejuk dimanupulasikan pada 0.0151 kg/s, 0.0161 kg/s dan 0.0168 kg/s. Maka, kertas kerja ini membentangkan mengenai simulasi pemindahan haba di dalam Shell and Tube Heat Exchanger Studies Unit (Model HE 667) yang digunakan di makmal kejuruteraan kimia UMP. Model CFD ini akan disahkan dengan membandingkan dengan keputusan eksperimen dalam lingkungan 15% ralat.

TABLE OF CONTENTS

CHAPTER	TITLE	PAGE
	ABSTRACT	i
	ABSTRAK	ii
	TABLE OF CONTENTS	iii
	LIST OF TABLES	vi
	LIST OF FIGURES	vii
	LIST OF SYMBOLS	ix
1	INTRODUCTION	1
1.1	Background of study	
1.1.1	Shell and Tube Heat Exchanger	1
1.1.2	Design of Shell and Tube Heat Exchanger	2
1.1.3	Heat Transfer	2
1.1.4	Computational Fluid Dynamics	3
1.1.5	Fluent	5
1.2	Problem Statement	7
1.3	Objective	7
1.4	Scopes of Research Project	7

2	LITERATURE REVIEW	8
2.1	Introduction	8
2.2	Computational Fluid Dynamics (CFD)	8
2.2.1	Definition	9
2.2.2	History	9
2.2.3	Simulation on Heat Transfer	11
2.2.4	Simulation Improving Efficiency	14
2.2.5	Graphical Portrait of Flow	16
2.3	Heat Exchanger Simulation	18
2.3.1	Tube and Finned Heat Exchanger	18
2.3.2	The Helical Baffle	20
2.3.3	Finned Tube and Unbaffled Shell and Tube Heat Exchanger	22
2.3.4	Tube in Tube Heat Exchanger	23
2.4	Steam Properties	25
2.5	Geometrical Description	25
3	METHODOLOGY	27
3.1	Introduction	27
3.2	Overall of Research Methodology	27
3.2.1	Experimental (HE 667)	29
3.2.1.1	Experimental Start-up Procedures	31
3.2.1.2	Experimental Procedure	31
3.2.2	Computational Fluid Dynamics	32
3.2.2.1	Problem Solving Step	32
3.2.2.2	Gambit 2.4	34
3.2.2.3	Geometry Modeling	34
3.2.2.4	Split Volume	35
3.2.2.5	Specifying the Target Volume	36

3.2.2.6	Meshing	37
3.2.2.7	Specifying the Volume	38
3.2.2.8	Specifying the Meshing Scheme	38
3.2.2.9	Cooper Meshing Scheme	41
3.2.2.10	Specifying Source Faces	42
3.2.2.11	Smooth Volume Meshes	43
3.2.2.12	Check Volume Meshes	43
3.2.2.13	Tabular 3-D Mesh Quality Data	44
3.2.2.14	Summarize Volume Mesh	46
3.2.2.15	Specifying Zone Types	46
3.2.2.16	Boundary Type Specifications	47
3.2.3	Fluent 6.3	48
3.2.3.1	Fluent Simulation Steps	49
4	RESULTS AND DISCUSSIONS	53
4.1	Introduction	53
4.2	Experimental Results	53
4.3	Simulation Results	54
4.4	Results Comparison and Validation	60
5	CONCLUSIONS AND RECOMMENDATIONS	62
5.1	Introduction	62
5.2	Conclusions	62
5.3	Recommendations	63
	REFERENCES	65
	APPENDICES	70

LIST OF TABLES

TABLE NO.	TITLE	PAGE
3.1	Valve Arrangements for Shell and Tube Heat Exchanger	32
3.2	Geometry Dimension for Shell and Tube Heat Exchanger	34
3.3	Geometry Specification for Shell and Tube Heat Exchanger	35
3.4	Element Scheme Specification	40
3.5	Summary of Smooth Volume Mesh	42
3.6	Check Volume Mesh Quality Tabular Output (Shell Side)	44
3.7	Check Volume Mesh Quality Tabular Output (Tube Side-Single Tube)	45
3.8	Check Volume Mesh Quality Tabular Output (Tube Side-37 Tubes)	45
3.9	Summary of Boundary Types Specification	48
3.10	Physical Properties of Borosilicate Glass	50
4.1	Counter-current Flow Results	54
4.2	Heat Transfer Rate Comparison (Hot Mass Flow Rate = 0.0158 kg/s)	60
4.3	Heat Transfer Rate Comparison (Hot Mass Flow Rate = 0.0161 kg/s)	61
4.4	Heat Transfer Rate Comparison (Hot Mass Flow Rate = 0.0168 kg/s)	61

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
1.1	Fluid Flow Simulation for a Shell and Tube Heat Exchanger	4
2.1	Photography of PF Tube Enwound with Helical Baffles	12
2.2	Graphical Portrait of Tube Exchanger	17
2.3	A Schematic of Heat Exchanger with Helical Baffles, Pitch Angle, and Baffle Space	20
3.1	Flowchart of Overall Methodology	28
3.2	Shell and Tube Heat Exchanger Study Unit (Model 667)	29
3.3	Schematic Diagram for Shell and Tube Heat Exchanger (Model 667)	30
3.4	Step of CFD Analyses	33
3.5	Split Volume Form	36
3.6	Mesh Volume Form	37
3.7	Smooth Volume Form	43
3.8	Specify Boundary Types Form	47
3.9	Program Structure	52
4.1	Scaled Residual Convergence after 3856 th Iterations	55
4.2	Contour of Static Temperature from Hot Water Inlet View	55

4.3	Contour of Static Temperature from Hot Water Outlet View	56
4.4	Contour of Static Temperature from Isometric View	57
4.5	Contour of Total Temperature from Isometric View	57
4.6	Graph of Total Temperature versus Distance (Linear Velocity of Hot Water = 0.0158 m/s)	58
4.7	Graph of Total Temperature versus Distance (Linear Velocity of Hot Water = 0.0161 m/s)	59
4.8	Graph of Total Temperature versus Distance (Linear Velocity of Hot Water = 0.0168 m/s)	59

LIST OF SYMBOLS

Φ_s	-	Pitch angle
H_s	-	Baffle space
OD	-	Outside diameter
ID	-	Inside diameter
C_p	-	Specific Heat
ΔT	-	Temperature difference
W	-	Heat Transfer Rate

CHAPTER 1

INTRODUCTION

1.1 Background of study

A shell and tube heat exchanger is a class of heat exchanger designs. It is the most common type of heat exchanger in oil refineries and other large chemical processes, and it is suitable for high pressure applications. As its name implies, this type of heat exchanger consists of a shell (a large pressure vessel) with a bundle of tubes inside the shell.

1.1.1 Shell and tube heat exchanger

The basic principle of operation is very simple as flows of two fluids with different temperature brought into close contact but prevented from mixing by a physical barrier. Then the temperature between two fluids tends to equalize by transfer of heat through the tube wall. The fluids can be either liquids or gases on either the shell or the tube side. In order to transfer heat efficiently, a large heat transfer area should be used, leading to the use of many tubes. In this way, waste heat can be put to use. This is an efficient way to conserve energy.

1.1.2 Design of shell and tube heat exchanger

There are several designs in shell and tube heat exchanger. Even though, the basic principle is still the same. The tubes may be straight or bent in the shape of a U, called U-tubes. This U-tubes type typically use in nuclear power plants. The heat exchanger is used to boil water recycled from a surface condenser into steam to drive a turbine to produce power. Most shell-and-tube heat exchangers are 1, 2, or 4 pass designs on the tube side. This refers to the number of times the fluid in the tubes passes through the fluid in the shell. In a single pass heat exchanger, the fluid goes in one end of each tube and out the other.

1.1.3 Heat Transfer

Heat transfer is the terms use for thermal energy from a hot to a colder body. Theoretically on a microscopic scale, thermal energy is related is related to the kinetic energy of molecules. The greater a material's temperature, the greater the thermal agitation of its constituent molecules. Then the regions containing greater molecular kinetic energy will pass this energy to regions with less kinetic energy. So when a physical body likes an object or fluid, is at a different temperature than its surroundings or another body, heat transfer will occurs in such a way that the body and the surroundings reach thermal equilibrium.

Heat transfer always occurs from a hot body to a cold one, a result of the second law of thermodynamics. Where there is a temperature difference between objects in proximity, heat transfer between them can never be stopped but can only be slowed down. Transfer of thermal energy can only occurs through three ways which is conduction, convection and radiation or any combination of that.

In my case of study which relate to shell and tube heat exchanger is only consist of heat transfer by conduction and convection.

1.1.4 Computational Fluid Dynamics

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The technique is very powerful to perform the millions of calculations required to simulate the interaction of fluids and gases with complex surfaces used in engineering.

CFD not just spans on chemical industry, but a wide range of industrial and non-industrial application areas such as:

- Aerodynamics of aircraft and vehicles.
- Combustion in IC engines and gas turbines in power plant.
- Loads on offshore structures in marine engineering.
- Blood flows through arteries and vein in biomedical engineering.
- Weather prediction in meteorology.
- Flows inside rotating passages and diffusers in turbo-machinery.
- External and internal environment of building like wind loading and heating or ventilation system.
- Mixing and separation or polymer moulding in chemical process engineering.
- Distribution of pollutants and effluents in environmental engineering.

Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger (Figure 1.1). Furthermore, motor vehicle manufactures now routinely predict drag forces, under-bonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

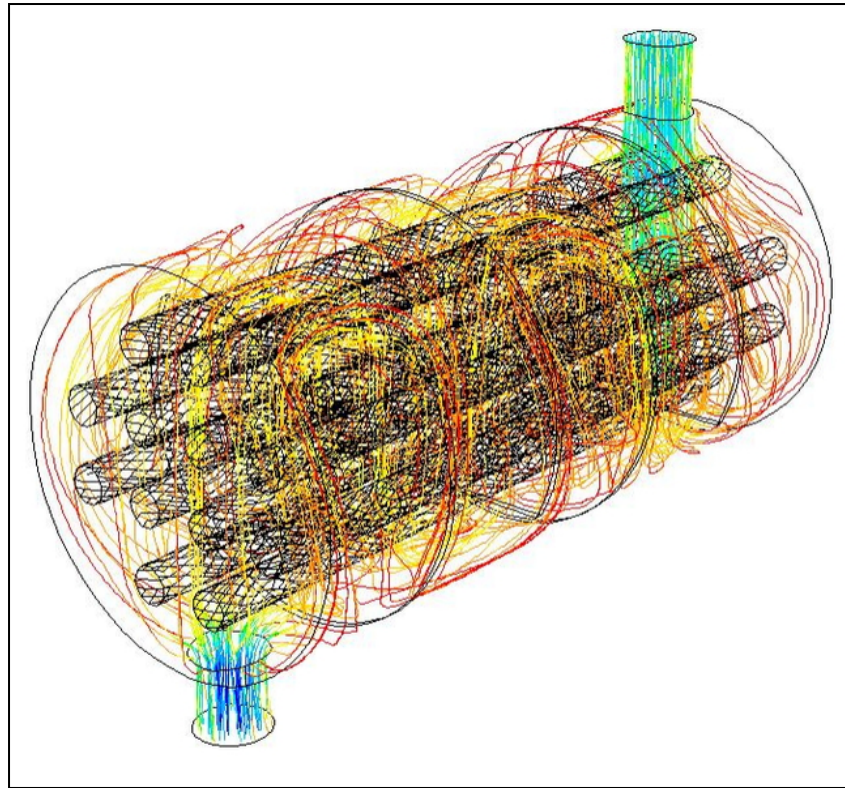


Figure 1.1: Fluid flow simulation for a shell and tube exchanger (Sadik et al. 2002).

The development in the CFD field provides a capability comparable to other Computer Aided Engineering (CAE) tools such as stress analysis codes. The availability of affordable high performance computing hardware and the introduction of user-friendly interfaces have led to a recent up surge of interest and CFD is poised to make an entry into the wider industrial community.

1.1.5 Fluent

Fluent is the world's largest provider of commercial computational fluid dynamics (CFD) software and services. fluent offers general-purpose CFD software for a wide range of industrial applications, along with highly automated, specifically focused packages. Fluent also offers CFD consulting services to customers worldwide. The staff at Fluent consists mostly of individuals with highly technical backgrounds as applied CFD engineers. In addition, fluent employs experts in computational methods, mesh generation, and software development.

Fluent's clients are the market leaders and the largest companies in industries such as automotive, aerospace, chemical and materials processing, power generation, biomedical, HVAC, and electronics.

Fluent is committed to furthering the body of knowledge on CFD, and to improving the effectiveness of computer modeling as a design and analysis tool in general. We invest in both internal research and development, and participate in collaborations with leading academic establishments, governments, and industry groups. We continue to explore and implement strategic alliances with both hardware and software providers to achieve greater synergy and efficiency for our customers.

Fluent's mission has been clear from the beginning: to work closely with customers to understand their fluid-flow challenges, to provide both software and services tailored to their needs, and to continually measure our success as a function of theirs. As a result of our continuing efforts to fulfill our mission, we have enjoyed outstanding user loyalty throughout our history.

Fluent is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. Fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated

about complex geometries with relative ease. Supported mesh types include 2D triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge/ polyhedral, and mixed (hybrid) meshes. Fluent also allows you to refine or coarsen your grid based on the flow solution.

Fluent is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, fluent uses a client/server architecture, which allows it to run as separate simultaneous processes on client desktop workstations and powerful compute servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems.

All functions required to compute a solution and display the results are accessible in fluent through an interactive, menu-driven interface.

1.2 Problem Statement

Heat transfer is considered as transfer of thermal energy from physical body to another. Heat transfer is the most important parameter to be measured as the performance and efficiency of the shell and tube heat exchanger. By using CFD simulation software, it can reduce the time and operation cost compared by experimental in order to measure the optimum parameter and the behavior of this type of heat exchanger.

1.3 Objective of research project:

The objective of this study is to develop a CFD simulation to predict heat transfer in shell and tube heat exchanger.

1.4 Scopes of research project:

The scopes of the research project are:

- To simulate heat transfer in shell and tube heat exchanger by using CFD-Fluent software.
- To analyze the heat transfer in shell and tube heat exchanger by comparing the simulation result to the experimental results.
- Validate simulation results to the experimental results within 15% error.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The purpose of this chapter is to provide a literature review of past research effort such as journals or articles related to shell and tube heat exchanger and computational fluid dynamics (CFD) analysis whether on two dimension and three dimension modeling. Moreover, review of other relevant research studies are made to provide more information in order to understand more on this research.

2.2 Computational Fluid Dynamics (CFD)

CFD is a computational technology that enables to study the dynamics of things that flow. CFD can build a computational model that represents a system or device. Then, by apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena.

2.2.1 Definition

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design (Fluent.com).

Computational fluid dynamics (CFD) is useful in a wide variety of applications and use in industry. The simulation is performed using the FLUENT software. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyze problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering.

2.2.2 History

Since the dawn of civilization, mankind has always had a fascination with fluids; whether it is the flow of water in rivers, the wind and weather in our atmosphere, the smelting of metals, powerful ocean currents or the flow of blood around our bodies.

In antiquity, great Greek thinkers like Heraclitus postulated that "Everything flows" but he was thinking of this in a philosophical sense rather than in a recognizably scientific way. However, Archimedes initiated the fields of static mechanics, hydrostatics, and determined how to measure densities and volumes of objects. The

focus at the time was on waterworks: aqueducts, canals, harbors, and bathhouses, which the ancient Romans perfected to a science.

It was not until the Renaissance that these ideas resurfaced again in Southern Europe when we find great artists cum engineers like Leonardo Da Vinci starting to examine the natural world of fluids and flow in detail again. He observed natural phenomena in the visible world, recognizing their form and structure, and describing them pictorially exactly as they were. He planned and supervised canal and harbor works over a large part of middle Italy. His contributions to fluid mechanics are presented in a nine part treatise that covers water surfaces, movement of water, water waves, eddies, falling water, free jets, interference of waves, and many other newly observed phenomena.

Significant work was done trying to mathematically describe the motion of fluids. Daniel Bernoulli (1700-1782) derived Bernoulli's famous equation, and Leonhard Euler (1707-1783) proposed the Euler equations, which describe the conservation of momentum for an inviscid fluid, and conservation of mass. He also proposed the velocity potential theory. Two other very important contributors to the field of fluid flow emerged at this time; the Frenchman, Claude Louis Marie Henry Navier (1785-1836) and the Irishman, George Gabriel Stokes (1819-1903) who introduced viscous transport into the Euler equations, which resulted in the now famous Navier-Stokes equation.

It is debatable as to who did the earliest CFD calculations (in a modern sense) although Lewis Fry Richardson in England (1881-1953) developed the first numerical weather prediction system when he divided physical space into grid cells and used the finite difference approximations of Bjerknes's "primitive differential equations". His own attempt to calculate weather for a single eight-hour period took six weeks of real time and ended in failure! His model's enormous calculation requirements led Richardson to propose a solution he called the "forecast-factory". The "factory" would

have involved filling a vast stadium with 64,000 people. Each one, armed with a mechanical calculator, would perform part of the flow calculation. A leader in the center, using colored signal lights and telegraph communication, would coordinate the forecast. What he was proposing would have been a very rudimentary CFD calculation.

CFD is now recognized to be a part of the computer-aided engineering (CAE) spectrum of tools used extensively today in all industries, and its approach to modeling fluid flow phenomena allows equipment designers and technical analysts to have the power of a virtual wind tunnel on their desktop computer. CFD software has evolved far beyond what Navier, Stokes or Da Vinci could ever have imagined. CFD has become an indispensable part of the aerodynamic and hydrodynamic design process for planes, trains, automobiles, rockets, ships, submarines; and indeed any moving craft or manufacturing process that mankind has devised (Fluent.com).

2.2.3 Simulation on heat transfer.

The shell-and-tube heat exchanger is widely used equipment in various industries such as process, power generation, petroleum refining, chemicals and paper. According to a market survey conducted in Europe, it accounts for about 42% of the market share. Energy and materials savings considerations, as well as environmental challenges in the industry have stimulated the demand for high efficiency heat exchangers.

The developments for shell-and-tube exchangers center on better conversion of pressure drop into heat transfer by improving the conventional baffle designs. A good baffle design, while attempting to direct the flowing a plug flow manner, also has to fulfill the main function of providing adequate tube support.

Helical baffle as one of novel shell side baffle geometries was developed to increase the efficiency of heat transfer. Although shell-and-tube heat exchanger with helical baffles appear to offer significant advantages over conventional exchanger with segmental baffles, very few studies of this type of heat exchanger could be found in the literature, in particular, on heat transfer enhancement and numerical simulation. Lutchu and Nemcansky (1990) found that helical baffle geometry could force the shell side flow field to approach a plug flow condition, which increased the average temperature driving force. The flow patterns induced by the baffles also caused the shell side heat transfer to increase markedly. Kral et al. conducted the hydrodynamic studies of the shell side on a helically baffled heat exchanger model made of Perspex using stimulus-response techniques.

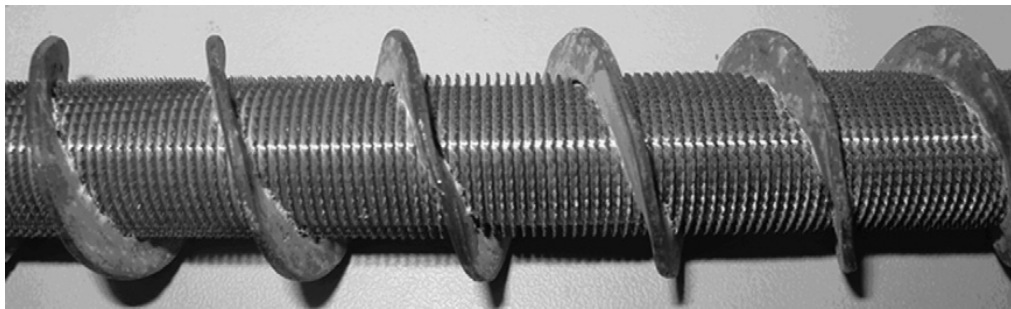


Figure 2.1: Photography of PF tube enwound with helical baffles
(Zhengguo et al. 2008)

The results showed that a helically baffled heat exchanger provided an ideal shell side geometry resulting in a uniform flow path with low degree of back mixing and nearly negligible dead volume. Performance of heat exchangers with helical baffles was discussed using the results of tests conducted on unit with various baffle geometries.

The researcher study on correction factors for shell-and-tube heat exchangers with segmental as compared to helical baffles. Chunangad et al. (1997) presented a case study on the industrial application of a helically baffled heat exchanger combined with

integral low finned tubes; the results showed that, at the same heat duty, both the equivalent bare tube surface and shell side pressure drop were reduced by one half of that required in the original segment baffled bare tube design for platform gas cooling with sea water.

Sivashanmugam and Suresh reported heat transfer enhancement for circular tube fitted with helical screw element of different twist ratio. The effect of spacer length on heat transfer augmentation and friction factor, and the effect of twist ratio on heat transfer augmentation and friction factor have been presented separately. Eiamsa-ard and Promvonge (2007) studied that the effects of insertion of a helical screw-tape with or without core-rod in a concentric double tube heat exchanger on heat transfer and flow friction characteristics are experimentally investigated. The results showed that helical screw element could enhance heat transfer. But in these literatures, all tested inner tubes are smooth tubes.

Although crucial for process design, the correlations offer little insight about the detailed flows that take place in the exchanger. Computational fluid dynamics (CFD) offers a convenient means to study the detailed flows and heat exchange processes that take place inside the shell. Andrews and Master (2005) had performed detailed three-dimensional CFD simulations to explore the performance of a helically baffled heat exchanger.

The CFD simulation shows a comparison with plug flow showed that the helically baffled heat exchanger had a fluid turn ratio of 0.64, 0.78 and 0.77 for the 10°, 25° and 40° helix angles, indicating more overall plug-like flow the higher helix angles. Computed pressure drops compare reasonably well with ABB Lummus Heat Transfer correlation results. Shen et al. (2004) established a mathematical model of the flow and heat transfer of the helical baffles heat exchanger with the theory of the Reynolds stress model applied.

The commercial software FLUENT is used to simulate the influence of helical baffles on heat transfer capability and flow resistance of helical baffle heat exchangers by unstructured grid. The numerical simulation results at the 35° helix inclination angle are compared with those of experiment. In previous papers, the heat transfer and pressure drop of helically baffled heat exchanger combined with petal shaped finned (PF) tubes for oil cooling with water as the coolant were investigated, and was compared with those of helically baffled heat exchanger combined with integral low finned tubes. The experimental results showed that, for the heat exchanger with PF tubes, the shell side heat transfer coefficients were augmented by 28–48%, whereas the shell side pressure drops were reduced by 35–75% at the same volumetric flow rates of oil.

In the current work, the experimental study and numerical simulation on heat transfer and pressure drop characteristics were performed at the shell side of a helically baffled heat exchanger combined with one PF tube. The flow field and heat transfer performances in the shell side were simulated using commercial fluent software. The numerical results of the shell side Nusselt number and pressure drop were compared with those of experimental data.

2.2.4 Simulation Improving Efficiency

CFD simulation helped increase the efficiency of a heat exchanger by a factor of nine by showing that wire matrix inserts would overcome a flow distribution problem. The customer consulted Cal Gavin because an installed 324 tube exchanger was only providing a fraction of its theoretical performance.

The subsequent simulation with computational fluid dynamics (CFD) software showed that nearly 70% of the exchanger pressure drop was lost across the nozzles. The remainder was insufficient to evenly distribute the fluid through the tube bundle.

Further analysis indicated adding inserts to increase the flow resistance of the bundle could solve the problem.

Cal Gavin is a process-oriented company of dedicated chemical engineering professionals whose primary charter is to deliver imaginative solutions to improve the performance and economics of processing fluids. Services provided by the firm include integrated plant reviews, design and simulation, CFD modeling, ability studies, troubleshooting, optimization, de-bottlenecking, team support, and process consultancy. Cal Gavin specialize in fluid contacting dynamics for combined heat and mass transfer, correction of fluid, fractional crystallization, reaction engineering, flow regime control and fluid mixing and dispersing. The company sponsors a progressive program of fundamental research both in-house and in universities in the UK and numerous countries worldwide. In the application described above engineers from a German chemical company contacted Cal Gavin with a problem which conventional approaches failed to resolve. The 700mm diameter, 2500mm long vertical exchanger had 200mm nozzles on the tube side. It was designed to operate with a tube side flow of 90757 kg/hr at 69.5C with the fluid density at 873 (kg/m³) and viscosity of 1cP. The heat exchanger was providing far less thermal duty than necessary for the application. The alternative solution to this problem would have been to replace the exchanger with a larger involving significant installation work.

Cal Gavin engineers recognized that in theory the heat exchanger should have been able to meet required duty and suspected a fluid flow distribution problem. While heat exchanger design typically focuses on surface area requirements the fluid flow within the tubes can be of equal importance. A shell and tube heat exchanger consists of a bundle of tubes through which one fluid flows whilst the other fluid flows around the tubes. This accommodates a large surface area in a given volume. The target is obviously equal distribution of flow in each tube. Should most of the fluid flow through just a few tubes the majority of the installed surface area is wasted.